



Analysis of Turbo Ventilator Using CFD

Mr. GANESH K. JADHAV
Research Scholar
Sinhgad College of Engineering
Pune

Dr. P. M. GHANEGAONKAR
Professor
Mechanical Engg. Dept.
Dr. D.Y. Patil Inst. of Engg. & Tech. Pimpri, Pune

Mr. SHARAD GARG
Design Engineer
Continuum Technology
Pune

Abstract— Rotating turbo-ventilators are cost effective environmental friendly natural ventilation devices, which are used to extract polluted air from a building. A Computational Fluid Dynamics (CFD) study using the standard $k-\epsilon$ turbulence model with multiple reference frames (MRF) meshing technique is used to explore the suitability of numerical approach in modeling various features of a ventilator flow. The initial CFD results were validated against experimental results carried out on the test rig. It is observed that application of CFD simulation is feasible as a cost effective tool in the future design, development and performance analysis of rotating wind driven turbo ventilators.

Keywords - CFD, Ventilation, Turbo ventilator

I. INTRODUCTION

Ventilation is very important to maintain humidity, condensation, overheating and remove odor, smokes and pollutant [1]. Use of the turbo ventilator is an easy, environmental friendly and economical way to gain good ventilation in industry. The turbo ventilator is always preferred choice to ventilate the buildings without depending on air conditioning system, which is usually associated with global warming. It was originally patented by Meadows in 1929 and is considered as one of the type of the roof ventilator, which has been defined by American Society of Heating, Refrigerating and Air-Conditioning Engineers (ASHRAE) in 1999. [2].

A turbo ventilator roof dome is mounted on top of the building. A turbo ventilator consists of number of vertical blades (it may be curved or straight blades) in a spherical array mounted on a frame. At the center, a shaft is supported by upper and lower bearings. When wind blows on the vanes the resulting aerodynamic drag forces cause the turbo ventilator to rotate. This rotation produces a negative pressure inside the turbo ventilator and air is thus sucked from the base duct. Air enters the turbine axially via the base duct and is expelled radially. [3] In the absence of wind, a turbine ventilator facilitates ventilation using stack effects. [4] Dale and Ackerman [5] observed that the flow rate of a turbine ventilator under field conditions is dependent on the wind speed and wind direction.

Design of the turbo ventilator appears simple but shape of the blade is very complicated [2][4] and has, therefore, It becomes the subject of several investigations.

Lai C. M. [6] [7], have conducted flow visualization and physical experiments in wind tunnels to compare

the performance of different types of rooftop ventilator at different Reynolds number flows.

The CFD results were validated against experimental results obtained for a commercial ventilator tested on test rig.

II. CFD MODELING

The numerical simulation of flow characteristics of the turbo ventilator is performed using STAR CCM+ 10.0. The equations for the conservation of mass momentum and turbulence scales are solved in STAR CCM+ 10.0. CFD studies [8] are largely dependent on the meshing quality, turbulence models, boundary conditions and difference schemes. STAR-CCM+ employs mathematical model of turbulence to determine the Reynolds stresses and turbulent scalar fluxes. Special models are also employed to characterize the flow in wall boundary layers. The turbulence models incorporated in the current version of STAR-CCM+ are listed below:

- i) $k-\epsilon$ Turbulence Model
- ii) $k-\omega$ Turbulence Model

A K-Epsilon turbulence model is a two-equation model in which transport equations are solved for the turbulent kinetic energy (k) and its dissipation rate ϵ [8]. Various forms of the K-Epsilon models have been in use for several decades, and the $k-\epsilon$ model is very popular for industrial applications due to its good convergence rate and relatively low memory requirements. It does not very accurately compute flow fields that exhibit adverse pressure gradients, strong curvature to the flow, or jet flow. It does perform well for external flow problems around complex geometries. For example, the k -epsilon model can be used to solve for the airflow around a bluff body.

Since the inception of the K-Epsilon model, there have been several attempts to improve it. The most significant of these improvements have been incorporated into STAR-CCM+.

STAR-CCM+ has a choice of different K-Epsilon turbulence models:

- Standard K-Epsilon
- Standard Two-Layer K-Epsilon
- Realizable K-Epsilon
- Realizable Two-Layer K-Epsilon

Non-linear constitutive relations are available in conjunction with the three standard K-Epsilon turbulence model variants. In present investigation Realizable Two-Layer K-Epsilon treatment model is used.

III. GEOMETRY AND BOUNDARY CONDITION

The geometry of turbo ventilator with number of blades is 42 and blade height 300 mm is created in Computer Aided Three dimensional Interactive Application (CATIA) and imported to STAR CCM+ 10.0 (the pre-processor) for meshing as shown in Fig.1. STAR CCM+ is used for meshing as well as simulation. For modeling of the rotating motion of the turbo ventilator, the multiple reference frames (MRF) meshing in STAR CCM+ is used. This meshing allows multiple moving reference frames to be solved in a single domain. MRF is normally used for steady state condition and the individual cell zone can move at different rotational speeds and provide solution using the moving reference frame equations. Grid sensitivity test was performed by examining the effect of different number of mesh grids and the total number of grids was determined when grid independence was established.

The coordinate system for the model is defined below the X axis defines free stream direction while Y axis points to XY and XZ planes respectively. The turbo ventilator rotates about the Z axis.

The turbo ventilator model is vertically centered with height 300 mm and throat diameter 600 mm and overall diameter of turbo ventilator is 720 mm. The wind tunnel dimension is 6480x 3600x 2880 mm.

For proper meshing and good resolution the realizable k- epsilon model is used. A fine mesh of 1 cm at the wall boundaries particularly at the blade surfaces of ventilator. For the room domain mesh base size used is 5 cm. The room domain is discretized for 10, 06,395 cells as shown in Fig. 2

Boundary conditions

Velocity Inlet

In STAR-CCM+ the boundary condition for velocity inlet, i.e. the inlet face velocity vector, is specified directly. The velocity is 5 m/s is taken for the

simulation as the average wind velocity is 5-6 m/s and temperature is atmospheric as shown in Fig. 3

Pressure Outlet

Pressure outlet specifies the pressure on the outlet boundary to be equal to zero. At the pressure outlet the boundary face velocity is extrapolated from the interiorizing reconstruction gradients.

Wall

By using this boundary condition, the tangential velocity at the boundary is set to zero. The boundary layer face pressure is extrapolated from the adjacent cells using reconstruction gradient. There will not be any velocities through a boundary defined as a wall.

Simulation time using 32 GB ram Dual core processor was approximately 750 Hrs.

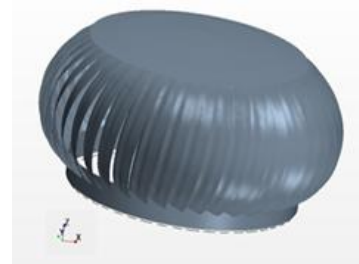


Fig. 1 3 D Model of Turbo Ventilator



Fig. 2 Meshed Model

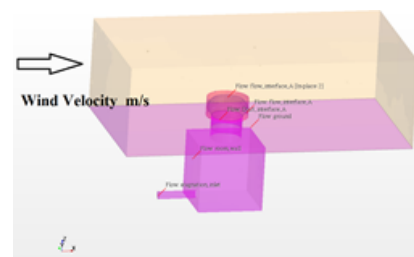


Fig. 3 Ventilator Boundary Condition

IV. RESULTS AND PLOTS

An eddy is the swirling of a fluid and the reverse current created when the fluid flows when it moves past an obstacle. The moving fluid creates an empty space of downstream side. Fluid behind the obstacle flows into the void creating a swirl of fluid on each edge of the obstacle, followed by a short reverse flow of fluid behind the obstacle flowing upstream, toward the back of the obstacle. Ventilator acts as an obstacle to the flow of air. So there is a formation of eddies as shown in Fig. 4

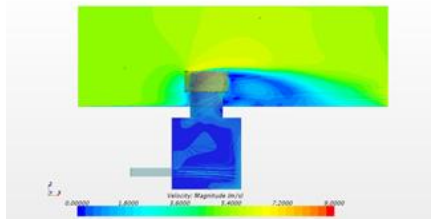


Fig. 4 Velocity Plot Of Ventilator Model

Vortex Formation within the Ventilator:

A vortex is a region within a fluid where the flow is mostly a spinning motion about an imaginary axis, straight or curved. Ideally it is vortex formation should be at the center of turbo ventilator having vertical imaginary axis. Due to oval shape of turbo ventilator the imaginary axis shifts toward the left hand side portion of the ventilator. Plot 5 and 6 shows the images of CFD results of turbo ventilator. It is observed that there is low pressure at the center of vortex, which increases as it moves away from the center of vortex.

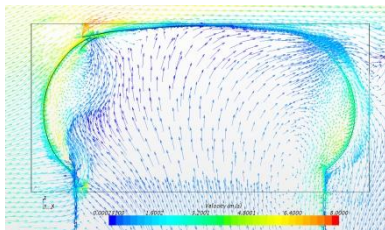


Fig. 5 Velocity Vector Contour (Elevation View)

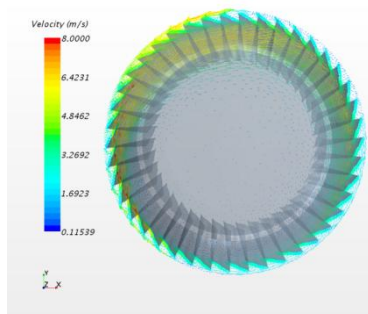


Fig. 6 Velocity Vector Contour (Plan View)

Vector Plot:

A vector plot shown in Fig.7 indicates direction of flow within and around the turbo ventilator. Vector quantity is displayed at discrete points (usually velocity, with arrows) whose orientation indicates direction and whose size indicates magnitude. It is observed that upward arrows of velocity vector indicate that stale air is thrown outside from the plenum chamber.

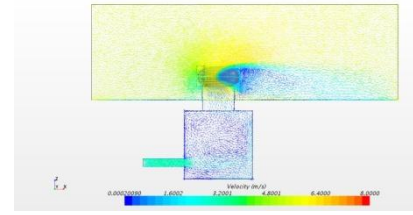


Fig. 7 Velocity Vector Plots

Pressure Difference Region:

Pressure inside the turbo ventilator is minimum, while pressure is maximum at incoming air side. When air is standstill condition and if ventilator is rotating, pressure is minimum at center of ventilator, but in actual practice air entry is from one side of the ventilator so the minimum pressure zone shifts towards the air entry side. Further at the back side of the ventilator pressure goes on decreasing due to eddies formation as shown in Fig. 8

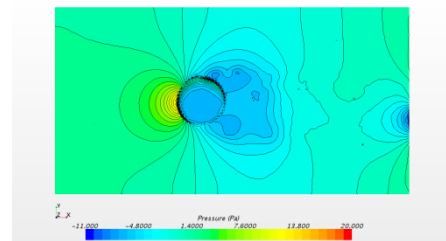
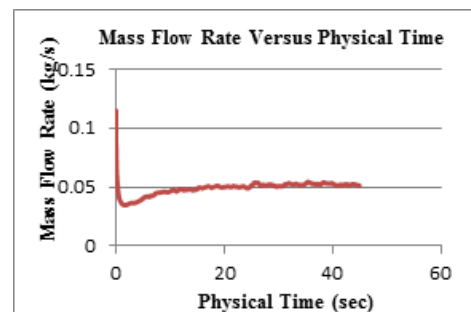


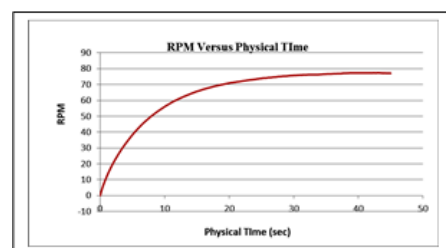
Fig. 8 Pressure Plot (Plan View)

CFD Results:



Plot 1 Mass Flow rate versus Wind velocity

It is clear from the plot 1 the simulation is converse and has researched to steady state condition after 40 second as physical time. Further concluded trial simulation is completed and corresponding mass flow rate is 0.05 kg/sec (40 lps). Plot 2 indicates that rotational speed of turbo ventilator is 78 rpm.



Plot 2 Rotational Speed versus Physical Time

1. Experimental Testing of Turbo Ventilator

Turbo ventilator is tested for its performance using experimental test rig and results obtained and experimental results were compared with CFD are shown in table 1

These results are validated with the results of the experiment readings are carried out on the test rig.

Table 1

Parameter	CFD Result	Experimental results	Percentage difference (%)
Wind Velocity (m/s)	5	5	-
Rotational Speed (rpm)	78	76	2.5
Mass flow rate (lps)	40	37	7.5

V. CONCLUSION

CFD simulation has been carried out to analyze the important flow features associated with the operation of a wind driven turbo ventilator. Standard k-ε turbulence model with multiple reference frames (MRF) meshing technique was used.

The CFD results of such complex modeling of rotating turbo ventilator are successfully simulated. Also it helped to study the internal as well as external flow around and in the turbo ventilator.

As wind impingement on the turbo ventilator increases with increase in the rotational speed results in increase in mass flow rate. CFD results are validated with the results of the experiment readings are carried out on the test rig with 5 % percentage difference in the mass flow rate and 2.5% difference in the rotational speed. CFD simulation is effective tool development and performance analysis of rotating wind driven turbo ventilator.

VI. REFERENCES

- [1]. Rashid DMH, Ahmed NA. Study of aerodynamic forces on a rotating wind
- [2]. Meadows, V. H. Rotary ventilator. US Patent 1, 857, and 762. (1929).
- [3]. Naghman Khan et al., A review on wind driven ventilation techniques Energy and buildings 2008; 40:1586-60
- [4]. Naghman Khan et al., "Performance of testing and comparison of turbine ventilators", Journal of Renewable Energy, 2008;33:2441-2447
- [5]. Dale, J. D., & Ackerman, M. Y. (1993). Evaluation of the performance of attic turbine ventilators. ASHRAE Transactions, 99(1), 14-22
- [6]. Kuo I. S., & Lai, C. M. Assessment of the potential of roof turbine ventilators for bathroom ventilation.
- [7]. Kuo I. S., & Lai, C. M. Assessment of the potential of roof turbine ventilators for bathroom ventilation. Building Services Engineering Research and Technology 2005; 26(2):173-179
- [8]. Shao-Ting J. Lien et al. "Numerical simulation of rooftop ventilator flow" Building and Environment 45 (2010) 1808-1815
- [9]. Lai CM. Experiments on the ventilation efficiency of turbine ventilators used for building and factory ventilation. Energy Build 2003; 35:927–32.
- [10]. Computational Fluid Dynamics by John D. Anderson, McGraw-Hill International editions.
- [11]. "Fluid Mechanics (Fundamentals and Applications)" by Yunus A. Cengel and John M. Cimbala (2nd Edition).